DYNAMICS OF FLAME STABILIZED BY TRIANGULAR BLUFF BODY IN PARTIALLY PREMIXED METHANE-AIR COMBUSTION

T. V. Santosh Kumar*

P. R. Alemela

J. B. W Kok

Department of Thermal Engineering (THW), School of Engineering (CTW), University of Twente, The Netherlands *Email: santosh.tarband@utwente.nl

ABSTRACT

In the design and operational tuning of gas turbine combustors it is important to be able to predict the interaction of the flame stabilization recirculation area with the burner aerodynamics. In the present paper transient computational fluid dynamics analysis is used to study these effects. Vortex interactions with the flame play a key role in many practical combustion systems. The interactions drive a large class of combustion instabilities and are responsible for changing the reaction rates, shape of the flame and the global heat release rate. The evolution of vortex shedding in reactive flows and its effects on the dynamics of the flame are important to be predicted. The present study describes dynamics of bluff body stabilized flames in a partially premixed combustion system. The bluff body is an equilateral wedge that induces the flame recirculation zone. The wedge is positioned at onethird length of the duct, which, is acoustically closed at the bottom end and open at the top. Transient computational modeling of partially premixed combustion is carried out using the commercial ANSYS CFX code and the results show that the vortex shedding has a destabilizing effect on the combustion process. Scale Adaptive Simulation turbulence model is used to compare between non-reacting cases and combustion flows to show the effects of aerodynamics-combustion coupling. The transient data reveals that frequency peaks of pressure and temperature spectra and is consistent with the longitudinal natural frequencies and Kelvin-Helmholtz instability frequency for reactive flow simulations. The same phenomenon is observed at different operating conditions of varying power. It has also been shown that the pressure and heat release are in phase, satisfying the Rayleigh criterion and therefore indicating the presence of aerodynamic-combustion instability. The data are compared to the scarce data on experiments and simulations available in literature.

Keywords: Methane-air combustion, bluff body, coupled and uncoupled modes, limit cycle oscillations, KH instability.

NOMENCLATURE

- Re Reynolds Number
- St Strouhal Number
- Le Lewis Number
- KH Kelvin Helmholtz
- BVK Bernard Von Karman
- *T_b Burnt Mixture Temperature*
- *T_u* Unburnt Mixture Temperature
- *T_{cc} Average Temperature in combustor*
- D Width of the wedge (25mm)
- *L Length of the combustor*
- λ Air excess ratio
- φ Equivalence ratio

INTRODUCTION

The flame in a gas turbine combustion chamber can only be stabilized over a range of operating conditions, determining these conditions, where the combustion system exhibits instability is important for the safe operation and longer life of the combustion system.

A wide variety of approaches can be used to stabilize the flame in the combustion chamber, bluff body stabilization is of particular interest here. Bluff bodies are used for flame stabilization in a variety of combustion and propulsion systems.

The physical phenomenon which places the flame over the bluff body involves the combustible mixture being in contact with the hot combusted products in the recirculation zone of the bluff body wake and continuous ignition in the shear layers bounding the recirculation zone. The global flame holder characteristics have been investigated by Santosh et al. [1] and Lieuwen et al. [2] which suggest that the bluff body flame stabilizes in a sequence of steps, depending on the density and temperature jumps in the flow (exothermic influences on the bluff body wake). Further they have also reported a distinctive sinuous vortex shedding for ϕ =0.62.

Furebuy et al. [3, 4] suggest that the strength of the large vortices, which appeared in cold flow, is decreased in the reacting case as a result of the change in the viscosity, which decreases the Reynolds Number. But such stabilizing effects occur only at high Reynolds Numbers and are greatly influenced by the blockage effects. Furebuy [4] had also performed LES on similar test case and suggested that the frequency of vortex shedding in the reacting case has shifted from 100Hz in cold case to 120Hz characterized by periodic vortex shedding from both sides of the bluff body.

Nicholson et al. [5] has also observed that for φ =0.62 and very high temperature jumps (T_b/T_u=5-6) the flame exhibits BVK instability which is expressed as periodic pressure and heat release fluctuations this has also been confirmed Hertzberg et al. [6].Then Erikson et al. [7] simulations demonstrates that for high Reynolds Numbers and gas expansion ratios, T_b/T_u <1.5 the flow changes from highly symmetric to sinuous vortex shedding.

Although many simulations and experiments have been performed on the bluff body flames there is an uncertainty in predicting the effects of combustion on the bluff body wake at lower Re and equivalence ratios, this is the motivation of the paper to try to explain these causes and effects in the above explained regimes.

INSTABILITY MECHANISM

This section reviews a number of case studies in which coherent structures in the flow have been known to play an important role in the instability process

Poinsot et al. [8] observed that the vortices arising from the inlet jets (containing reactants) in the combustion chamber were involved in causing instability. The flame interacted with these vortices, grew in size and then broke up leading to a sudden maximum heat release. Also similar trends were observed experimentally see [19 - 21]

Schadow & Gutmark [9] reviewed literature involving coherent structures and suggested these were the potential drivers for combustion instability. They connected the structures arising in the flow to the inherent hydrodynamics instability arising from fluid dynamics. This can lead to certain configurations where the inherent instability frequency can happen to coincide with the "preferred mode" of the flow or the instability of any shear layer or any of the fundamental modes of the system.

Yu et al. [10] observed combustion instabilities in a dual-dump combustor at frequencies that were *not* connected to any of the eigenmodes of the combustor. The frequency at which the combustor was rendered unstable was actually a function of both acoustic time and vortex propagation time.

Cohen and Anderson [11] reported measurements carried out in a single-dump combustor. They found three principle frequencies, two of which were the longitudinal frequencies of the combustor. Ghoniem [12] carried out numerical and theoretical work on this configuration and attributed the third peak to be very unstable and suggested that the acoustic field was acting as an amplifier to excite this frequency.

Therefore there are a number of ways by which the coherent structures in the flow can lead to instability. But however a common observation can be made from all the studies that the vortical structures in the flow perturbs the flame; the flame responds to these perturbations giving rise to heat-release perturbation. This heat-release perturbation, under certain favorable conditions trigger periodic waves that gives rise to fresh vorticity oscillations and forms a closed loop.



Figure 1: Interaction diagram during self-excited oscillations involving vortices

To test the theories at hand and to understand how that instability mechanism develops into a limit-cycle a generic combustor has been constructed the details of which are given in figure 2.

COMBUSTOR DESIGN

The model combustor, which is used for CFD analysis, is described in Figure 2: Schematic of the generic combustor The dimensions are given in Table 1. It is a Rijke tube configuration, containing a triangular bluff body placed at around $1/3^{rd}$ of the length of the combustor. Around the wedge the tube is widened perpendicular to the wedge, in order to allow for a drop in flue gas density downstream the flame front. Partially premixed type of combustion occurs when the air injected far upstream mixes with fuel, injected from the side surfaces of the wedge (39 holes on each side) and the mixture is ignited. Fuel being used here is methane (CH₄) at room temperature (no preheating). The ignition takes place a few millimetres above the surface of the wedge where the flame stabilizes on the wedges wake.



Figure 2: Schematic of the generic combustor

Height upstream	360 [mm]
Height downstream	1070 [mm]
Upstream injection velocities	4.53,3.02,1.51 [m/s]
Cold flow Reynolds Number _{max}	6673
Combustion Power	60,40,20 [kW]
Width upstream & downstream	25,50 [mm]

Table 1: Geometric and flow details of the generic combustor

The combustor is 220 [mm] wide in the third direction along the length of the bluff body.

NUMERICAL SETUP

Time dependent 3-dimensional CFD analysis is carried out using ANSYS CFX v12.1. The geometry, which was described in combustor design, is used, with air inlet at the lower end of the combustor. Mass flow of gas is equally distributed over the 78 holes of injection located at two sides of the wedge. Velocity is specified for the air inlet and mass flow as boundary condition for the fuel inlet. Pressure outlet boundary condition is chosen for the outlet. Three different cases are considered for unsteady simulations as shown in Table 2: Test cases for reacting flows with varying power and fixed air excess ratio $\lambda = 1.60$

	Power [kW]	Inlet air velocity (V) [m/s]	Gas mass flow rate x 10 ⁻⁴ [kg/m ³]
Case 1	60	4.53	10.9
Case 2	40	3.02	7.27
Case 3	20	1.51	3.64

Table 2: Test cases for reacting flows with varying power and fixed air excess ratio $\lambda\,{=}1.60$

The non-reacting cases are also considered; they are similar to reacting cases, but with combustion model turned off. The walls are adiabatic for both non-reacting and reacting flows.

The computational domain was meshed with a minimum cell length less than *Imm* and the mesh contains around 950000 elements. URANS simulations are carried out with a time stepping $\Delta t = 10^{-5}$ s. The CFL number is kept below 1 to ensure convergence, accuracy and stability of the solution. The data is recorded from the URNAS simulations at every 100 time steps thus making a sampling frequency of 1000Hz. This data with frequency of 1000Hz is used to construct the FFT plots.

Since fuel and oxidizer (air) is inserted in to the domain at separate locations, therefore they are not perfectly mixed prior to combustion. For this reason the PDF flamelet model is used for combustion process, as it is a good and fast approach to solve non-premixed flames. The details of the model are discussed in the next section.

Large flow instability is observed in the flow region around the wedge, as a result of the shear layer instability and vortex shedding from the sharp edges of the bluff-body. This unstable flow has to be resolved well to get a better prediction of the overall combustion process. The turbulence model used here is the SAS-SST model, which is a URANS type model but with LES like behavior. It is proven to resolve unstable flows very well.

Apart from the cases described in Table 2, an open flame type of simulation is done with the same configuration but the downstream (after the wedge) width is increased from 50mm to 200mm. This case is compared to the experiments done at the University of Twente. A Photomultiplier tube is used to obtain the heat release rate and pressure transducers are used to measure dynamic pressure both inside the combustor and in the open-field. These measurements are used to validate the flame model for reacting simulations.

With adequate accuracy and sample data i.e., pressure, temperature and velocity fields are analyzed for any unique interactions, which are expressed as a consequence of thermo-acoustic instability.

GOVERNING EQUATIONS

The set of equations solved by ANSYS CFX are the unsteady Navier-Stokes equations in their conservative form. For all the equations static thermodynamic properties are specified unless otherwise specified.

Transport Equations

In this part the instantaneous mass momentum and energy equations are presented. For turbulent flows, the instantaneous equations are averaged leading to additional terms.

The instantaneous mass momentum and energy conservation can be written as following in a stationary frame: *Continuity equation*

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho U) = 0$$

Momentum equations

$$\frac{\partial(\rho U)}{\partial t} + \nabla . \left(\rho UU\right) = -\nabla p + \nabla . \tau + S_M \tag{2}$$

$$\tau = \mu(\nabla U + (\nabla U)^T - \frac{2}{2}\delta\nabla . U)$$
(3)

Total energy equation

$$\frac{\frac{\partial(\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla . \left(\rho U h_{tot}\right) = \nabla . \left(\lambda \nabla T\right) + \nabla . \left(U.\tau\right) + U.S_{M} + S_{E}$$
(4)

where h_{tot} is the total enthalpy, related to the static enthalpy h (T, p) by:

$$h_{tot} = h + \frac{1}{2}U^2 \tag{5}$$

The Navier-Stokes equations have the full capability to describe both turbulent and laminar behavior without need of additional information. However, turbulent flows at realistic Reynolds numbers span a large range of turbulent length and time scales, and would generally involve length scales much smaller than the smallest finite volume mesh, which cannot be practically used in all numerical analysis. Thus we make use of *CFD Turbulence models*.

Scale Adaptive Simulation (SAS-SST Model):

The Scale-Adaptive Simulation (SAS) is an improved URANS formulation for turbulence [13], which allows better resolution of the turbulent spectrum in unstable flow conditions. The SAS concept is based on the introduction of the von Karman length-scale into the turbulence scale equation. The information provided by the von Karman length-scale allows SAS models to dynamically adjust to

(1)

resolved structures in a URANS simulation, which results in a LES-like behavior in unsteady regions of the flow field. At the same time, the model provides standard RANS capabilities in stable flow regions.

Formulation:

The starting point of the transformation to the SST model is the *k*- v_t formulation as given by Menter et al. [13]. The following equations have been derived there for the variables k and $\varphi = k^{1/2}L$:

$$\begin{aligned} \frac{\partial\rho k}{\partial t} &+ \frac{\partial U_j \rho k}{\partial x_j} = P_K - c_\mu^{3/4} \rho \frac{k^2}{\Phi} + \frac{\partial}{\partial x} \left[\frac{\mu t}{\sigma_k} \frac{\partial k}{\partial y} \right] \quad (6) \\ \frac{\partial\rho \Phi}{\partial t} &+ \frac{\partial U_j \rho \Phi}{\partial x_j} = \zeta_1 \frac{\phi}{k} P_K - \zeta_2 \mu_t S |U''| \frac{\Phi^2}{k^{3/2}} - \zeta_3 \rho k + \frac{\partial}{\partial y} \left[\frac{\mu t}{\sigma_{\phi}} \frac{\partial \Phi}{\partial y} \right] \end{aligned} \tag{7}$$

with

$$|U''| = \sqrt{\frac{\partial^2 U_i}{\partial x_j^2} \frac{\partial^2 U_i}{\partial x_j^2}} \tag{9}$$

Where S is the absolute value of strain rate, P_k is the production rate of turbulent kinetic energy and $c_{\mu} = 0.09$, k = 0.41.

This SAS-relevant term in the equation for Φ is the term with second derivative |U''| as a result the length scale predicted by the above model is largely proportional to the von Karman length scale:

$$L_{\nu K} = K \left| \frac{\partial U/\partial y}{\partial^2 U/\partial y^2} \right|$$
(10)

The SAS –SST turbulence model has significant improvement over other RANS models. In an earlier study the k- ω SST turbulence model was used for the similar case. The differences in the results from the two models is presented below



Figure 3 Comparison between k-ω SST turbulence (a) and SAS-SST turbulence model (b) model showing pressure evolution in the CFD domain

The SAS-SST turbulence resolves the pressure much more accurately than other RANS models. We can see that the pressure evolves and forms a limit cycle in figure 3(b) which is not the case for $k-\omega$ SST turbulence model where the

pressure is just cyclic. It is also to be noted that correct values of pressure recorded in figure 3(b) which is close to experimental values.

Combustion Model: The PDF Flamelet Model

The Flamelet concept [14] for non-premixed combustion, describes the interaction of chemistry with turbulence in the limit of fast reactions (large Damköhler number). The combustion is assumed to occur in thin sheets with inner structure called Flamelets. The turbulent flame itself is treated as an ensemble of laminar Flamelets that are embedded into the flow field. The main advantage of the Flamelet model is that even though detailed information of molecular transport processes and elementary kinetic reactions are included, the numerical resolution of small length and time scales is not necessary. This avoids the wellknown problems of solving highly nonlinear kinetics in fluctuating flow fields and makes the method very robust. Only two scalar equations have to be solved independent of the number of chemical species involved in the simulation. Information of laminar model flames are pre-calculated and stored in a library to reduce computational time. On the other hand, the model is still restricted by assumptions like fast chemistry or the neglecting of different Lewis numbers of the chemical species.

The following list outlines the assumptions made to derive the Flamelet model:

- 1. Fast Chemistry
- 2. Unity Lewis numbers for all species, (Le = 1)
- 3. Combustion is in the Flamelet Regime
- Two feed system, i.e., fluid composition at boundaries must be pure "fuel," pure "oxidizer" or a linear blend of them.

Due to limitations of the code and hardware (computational expense) the simulations have been carried out with the help of PDF Flamelet combustion model. However the flamelet model well suited for combusting flows where the fuel and oxidizer are introduced as separate streams, which is the case here. Literature suggests [14] accurate predictions of turbulent combustion with PDF Flamelet model along with k- ϵ turbulence model. Also the flow Reynolds number in present case is in the turbulent regime, the fully developed turbulent flow ensures proper homogenous mixing of all the species in the flow. Thus the assumption of unity Lewis number may not make a big impact on the calculations. For methane-air combustions many of the intermediate species have a Le=1, which does not violate the assumption made by the combustion model to a great extent.

The assumptions made in deriving the Flamelet model may not be entirely suitable for studying the dynamic flame behavior; the effects of different models have to be investigated in the future.

Formulation of the energy equation

$$\rho \left(\frac{\partial T}{\partial \tau} + u_2 \frac{\partial T}{\partial Z_2} + u_3 \frac{\partial T}{\partial Z_3}\right) - \frac{\partial(\rho D)}{\partial x_2} \frac{\partial T}{\partial Z_2} - \frac{\partial(\rho D)}{\partial x_3} \frac{\partial T}{\partial Z_3} - \rho D \left[\left(\frac{\partial Z}{\partial x_\alpha}\right)^2 \frac{\partial^2 T}{\partial Z^2} + 2 \frac{\partial Z}{\partial x_2} \frac{\partial^2 T}{\partial Z \partial Z_2} + 2 \frac{\partial Z}{\partial 3} \frac{\partial^2 T}{\partial Z \partial Z_3} + \frac{\partial^2 T}{\partial Z_2^2} + \frac{\partial^2 T}{\partial Z_3^2} \right] = \frac{1}{\rho c_p} \sum_{i=1}^n h_i m_i + \frac{q_i}{c_p} + \frac{1}{c_p} \frac{\partial P}{\partial t}$$
(11)

Where mixture fraction Z is;

$$Z = \frac{Z_{Fuel}}{Z_{Fuel,1}} = 1 - \frac{Z_{Oxidiser}}{Z_{Oxidiser,2}}$$
(12)

Flamelet Libraries: The flamelet libraries provide the mean species mass fractions (Y_i) as a function of mean mixture fraction, variance of mixture fraction and scalar dissipation rate:

$$\tilde{Y}_i = \tilde{Y}_i(\tilde{Z}, Z^{"2}, \tilde{\chi}_{st})$$
⁽¹³⁾

ANSYS CFX has such pre-calculated libraries for methaneair combustion at STP.

RESULTS

Non-Reacting flows

To determine the exothermic effects on the flow in this configuration non-reacting simulations are carried out first. The 40kW case is considered with combustion model turned off. Transient simulation is carried out with the same numerical setup as the reacting flows. The Figure 4 shows normalised pressure spectrum at location 2D along the mid plane of the combustor, where D is the diameter of the wedge (25mm). The pressure spectrum reveals one peak frequency at about 30Hz. The peak observed here can be described as the frequency of vortex shedding over the bluff body. Based on this frequency f and the inlet velocity U, the Strouhal number (St=f D/U) is calculated to be 0.29. This is consistent with the literature [17], which also suggests that the value of the St is independent of the Re for the ranges of 50<Re<200,000 for non-reacting flows.

Brandon et al [17] have done numerical investigation of particle flow past square cylinder in a channel and found the St=0.2 for similar Reynolds numbers and blockage ratios.

Roshko et al. [15] observed a St=0.23 for cylindrical bluff body and St=0.28 for arbitrary bluff body shapes. This is base line for further simulations, the turbulence model is chosen based on this simulation.



Figure 4: FFT-pressure spectrum of non reacting simulation at U=3.01m/s similar to case 2

Reacting Cases

Open-flame

The results from the open-flame experiments are discussed first. The pressure and velocity spectrum from the numerical simulations are shown in figure 5. The spectrum shows that large-scale oscillation at low frequencies and less energy distribution at higher frequencies. However around 320Hz a significant peak in the velocity spectrum (see figure 5(a)) is observed this is consistent with the dip in pressure spectrum at the same frequency (see figure 5(b)). This frequency is believed to be present as a result of the Kelvin-Helmholtz (KH) instability, which is a function of BVK instability

 $f_{KH} = 0.023 f_{BVK} Re^{0.67}$

The KH instability is high frequency instability in velocity and pressure. They are expressed as small scale fluctuations in the flow field with lower amplitudes. Figure 7 shows the velocity contours plotted at frequency of 320Hz over one time period of 3.1ms. Notice that the inner recirculation zone at 0° and 360° have similar structure (coloured red in the contours). This shows that the frequency observed in the FFT plots (see figure 5) does belong to the small scale KH instabilities. These Kelvin-Helmholtz vortices some time stretch the flame front long enough to cause blow-off as reported by Chaudhuri et al [18]

(14)



Figure 5: FFT-pressure (a) and velocity (b) spectrum for open-flame simulation at 40kW and λ =1.60



Figure 6: FFT of the pressure signal from open field mic (a) and OH*-Chemiluminescence signal (b) measured at 40kW and λ =1.60



Figure 7: Instantaneous Velocity contour plots within a time period at different phase angles corresponding to KH instability (320 Hz), for open flame simulation at 40kW and λ =1.60.

From figures 5 and 6 we can see that the pressure spectrum from both simulations and experiments are quite comparable to each other. Similar trends are observed in the FFT of the pressure signal in figures 5(a) and FFT of the open-field microphone in figure 6(a). However the higher frequency of 320Hz is not visible in the experiments. The combustion and turbulence models thus behave well for both non-reacting and reacting cases.

Ducted flame

In this section the reacting flow simulations performed on the model combustor are discussed. Steady state analyses are carried out and converged solution (convergence criteria 10^{-5}) is used as initial conditions for the transient runs. In figure 8(a) the pressure versus time is plotted at 2D location in the mid plane of the combustor for 20kW and λ =1.60 where D (25mm) is the width of the wedge. The plots show that over

time the pressure rises and falls several times. This a typical instability inside combustion chamber where pressure fluctuations are behaving a cyclic manner which are caused due to hydrodynamics or thermo-acoustic instability. The FFT analysis of this data in figure 8(b) reveals that the pressure peaks at frequency 107Hz with a maximum amplitude. To see the effects of low amplitude oscillations at other frequencies the plots from this section onwards are plotted using Sound Pressure Level [dB].

As the power is doubled to 40kW with the same air excess ratio we observe almost the same frequency of 119Hz, this can be seen from the plots in the figure 9(b). It can be observed that for the 40kW case pressure raises very quickly inside the combustor seen in figure 9(a). Also at power 60KW and λ =1.60 we get the similar peak frequency of 110Hz as shown in Figure 10.



Figure 8: Time (a) and SPL-FFT- spectrum (b) of pressure signal from transient simulation at location 2D from the wedge in the mid plane of the combustor for 20kW and λ =1.60 showing a peak frequency of 107 Hz



Figure 9: Time (a) and SPL-FFT- spectrum (b) of pressure signal from transient simulation at location 2D from the wedge in the mid plane of the combustor for 40kW case and λ =1.60 showing peak frequency of 119Hz



Figure 10: Time signal (a) and SPL-FFT- spectrum (b) of pressure signal from transient simulation at location 2D from the wedge in the mid plane of the combustor for 60kW case and λ =1.60 showing peak frequency of 110Hz

This frequency of around 110Hz is not the vortex shedding frequency but is indeed close to the natural frequency of the combustor.

Acoustic Boundary Condition

For all the three cases of power 20, 40 and 60KW, the same frequency of ~110Hz is observed from the simulations, if we consider the current configuration to act as closed-open boundary condition and according to the boundary conditions set by ANSYS CFX (the continuous mass flow into the system at the air inlet acts as closed acoustic boundary condition and the pressure outlet boundary condition acts as open boundary condition, hence closedopen boundary condition, then the equation of calculating the fundamental frequency can be written as

$$f_n = (2n-1)c/4L$$
 (15)

Where c=694m/s is speed of sound calculated with average temperature in the combustor $T_{cc}=1200K$, and L=1.43m the total length of the combustor. According to this equation the *first Eigen mode of the combustor is around* 120Hz. The frequencies observed in the simulations are very close to this value. Therefore by varying the power we

always end up exciting the first Eigen-mode of the combustor.

The test rig at University of Twente has acoustically closed end at the air inlet and acoustically open condition at the exhaust. Similar boundary was expected in the simulations. However in ANSYS CFX there are no separate parameters to define reflective boundary conditions. Therefore from the transient analysis we try to reconstruct the pressure and velocity field in the CFD domain.



Figure 11: Locations of the pressure monitor points in the midplane upstream and downstream of the wedge

The pressure is monitored at several locations along the length of the combustor both upstream and downstream. And the data from these points is used to reconstruct pressure and velocity wave.



Figure 12: Pressure and velocity wave reconstruction with phase information

The pressure and velocity wave have been reconstructed in figure 12(a) & 12(b), we can see a quarter wave modes in both pressure and velocity, which conforms the open-closed acoustic boundary condition. And the constant phase in both pressure and velocity suggests that the wave captured is a standing wave. This kind of wave reconstruction from transient CFD data has not been reported in the literature.

In the case of 40kW & 60kW power a frequency of about 237Hz is observed. This frequency is not present in low power mode and is assumed to be associated with the acoustic natural frequency of the inlet duct i.e., the bottom part of the duct below the wedge. By using the same analogy of open-closed acoustic boundary condition for the duct and by using Eqn. 15 we get the acoustic natural frequency of the bottom duct (with 0.36 m long and c= 343 m/s) to be 238Hz. This frequency is consistent with the frequency that is recorded in Figures 9(b) & 10(b). This behaves as uncoupled mode corresponding to the inlet chamber and is mainly due to presence of non-linearities during limit cycle oscillations as reported recently by Sterling et al [26] and Sujith [27]. This kind of behavior can also be an effect of the acoustic and chemical source term as reported by Kok [28].

Also the third peak observed in the pressure data at ~ 450Hz for all three cases can be corresponded to the Kelvin-Helmholtz Instability which is a function of the vortex shedding frequency and it is of the order of 420Hz for present cases. It can also be argued as that this frequency 450 Hz is not the second Eigen mode of the combustor because for closed-open acoustic condition the second mode is around 365Hz and for open-open condition the second Eigen-mode is 500Hz. Hence this frequency may be associated to Kelvin-Helmholtz instability.

Therefore it can be seen that combustion instability in this particular configuration is self-excited and the system reaches instability. To measure the degree of instability in the system the Rayleigh criteria has to be studied. To do this the cross correlation between pressure and temperature (analogous to heat release rate) generally gives the overview of the feedback process between the combustion, aerodynamics and acoustics. In figures 13, 14 & 15 cross correlation amplitudes and phase are plotted for two cases. For the reacting cases the amplitude plots indicate that peak at around frequency of ~110Hz. This suggests the potential of the thermo-acoustic coupling in the system. And also in all the cases the Rayleigh criterion is satisfied with phase difference less than $\pi/2$ between pressure and heat-release rate see figure 13(b), 14(b) & 15(b).



Figure 13: Amplitude (a) and Phase (b) values of the Cross spectrum between temperature and pressure field at a location 2D downstream of the wedge in the mid-plane of the combustor for 20kW and λ =1.60



Figure 14: Amplitude (a) and Phase (b) values of the Cross spectrum between temperature and pressure field at a location 2D downstream of the wedge in the mid-plane of the combustor for 40kW and λ =1.60



Figure 15: Amplitude (a) and Phase (b) values of the Cross spectrum between temperature and pressure field at a location 2D downstream of the wedge in the mid-plane of the combustor for 60kW and λ =1.60

Rayleigh criterion delineates regimes of unstable combustion from the stable ones, but this is only a necessary condition that could be satisfied by a variety of combination of physical mechanisms

CONCLUSIONS

In the present paper combustion instabilities occurring by wedge stabilized flames are investigated. Transient high resolution CFD analysis using ANSYS-CFX is carried out. The system indicates that the pressure and heat release rate (temperature field in this case) are highly coupled and the system can reach limit cycle behaviour without any external excitation. To validate the turbulence and combustion models available with the ANSYS CFX code open flame experimental data has been used.

Three distinct frequencies have been found in the ducted flame simulations: the first at about 110Hz (lower), second 238Hz (intermediate) and third 450Hz (higher). The lower and higher frequency has been associated with the acoustic natural frequency of the total combustor (coupled) and Kelvin-Helmholtz instability respectively. Whereas the intermediate frequency is assumed to be uncoupled mode corresponding to the inlet chamber and is mainly due to presence of non-linearities during limit cycle oscillations. This is consistent with the existing literature.

The Rayleigh criteria have been verified in all three cases using the simulated pressure and temperature field. The phase delay between the heat release rate (indicated by temperature) and the pressure shows a value below 90° and thus suggests the system has the tendency to reach limit cycle behaviour.

ACKNOWLEDGMENTS

The work in this paper was performed with the support of the EC Marie Curie Action Initial Training Network program in project LIMOUSINE, grant application number 214905. The authors used ANSYS CFX for their simulations and thank Dr. Phil Stopford for his support.

REFERENCES

- Santosh, J. Shanbhogue, Sajjad, Husain, Tim, Lieuwen, Lean blowoff of bluff body stabilized flames: Scaling Dynamics, Progress in Energy and Combustion Science. 35(98-120), 1996
- [2] Tim Lieuwen, Santosh, J., Shanbhogue, Sachin Khosla and Clifford Smith, *Dynamics of bluff body flames near blowoff*, 45th AIAA Sciences Meeting and Exhibit, 2007
- [3] Fureby, C., Lofstorm, C., Large-Eddy Simulation of bluff body stabilized flames, 25th Symposium of The combustion Institute, 1994
- [4] Furebuy, C., A computational study of combustion instabilities due to vortex shedding, Proceedings of the Combustion Institute Vol.28/pp 783-79,1 2000
- [5] Nicholson, H. and Field, J., "Some Experimental Techniques for the Investigation of the Mechanism of Flame Stabilization in the Wake of Bluff Bodies," Proceedings of the Combustion Institute, Vol. 3, pp. 44-68, 1951

- [6] Hertzberg, J. R., Shepherd, I. G., and Talbot, L., Vortex Shedding Behind Rod Stabilized Flames, Combustion and Flame, Vol. 86, pp. 1-11, 1991
- [7] Erickson, R. R., Soteriou, M. C. and Mehta, P. G., *The influence of Temperature Ratio on the Dynamics of Bluff Body Stabilized Flames*, 44th Aerospace Sciences Meeting and Exhibit, Reno, Nevada, AIAA Paper 2006-753
- [8] Poinsot, T. J., Trouve, A. C., Veynante, D. P., Candel, S. M. and Esposito, E. J, Vortex-Driven Acoustically Coupled Combustion Instabilities, J. Fluid Mech. 177, 265-292, 1987
- [9] Schadow, K. C. and Gutmark, E, Combustion Instability Related to Vortex Shedding in Dump Combustors and Their Passive Control, Progr. Energy Combust. Sci. 18, 117-132, 1992
- [10] Yu, K. H., Trouve, A. and Daily, J. W, Low-Frequency Pressure Oscillations in a Model Ramjet, Combustor, J. Fluid Mech. 232, 47-72, 1991
- [11] Cohen, J. M. and Anderson, T. J, Experimental Investigation of near-Blowout Instabilities in a Lean, Premixed Step Combustor, AIAA Aerospace Sciences Meeting and Exhibit AIAA Paper No. 96-0819, 1996
- [12] Ghoniem, A. F., Annaswamy, A., Wee, D., Yi, T. and Park, S. Shear flow-driven combustion instability: evidence, simulation, and modeling. Proc. Combust. Inst. 29, 53-60, 2002
- [13] Menter F., R., Kuntz M., Bender R, A scale-adaptive simulation model for turbulent flow predictions. AIAA Paper 0767, 2003
- [14] Veynante, D., Vervisch, L., Turbulent combustion Modeling, Prog. in Energy and Comb. Sci.28, 193-266, 2002
- [15] Roshko, A., "On the drag and shedding frequency of two-dimensional bluff bodies", NACA Report No. 3169, 1954
- [16] Prasad, A. and Williamson A., The instability of shear layer separating from a bluff body, J. Fluid Mechanics. pp 375-402, 1997
- [17] D. J. Brandon and S. K. Aggarwal, A Numerical Investigation of Particle Deposition on a Square Cylinder Placed in a Channel Flow, Aerosol Science and Technology 34: 340–352, 2001.
- [18] S. Chaudhuri, S. Kostka, M. W. Renfro, B. M. Cetegen, *Blowoff dynamics of bluff body stabilized* conical premixed flames. Combustion and Flame, Volume 157, Issue 4, April 2010, 790-802
- [19] S. K. Aggarwal, Structure of Unsteady Partially Premixed Flames and the Existence of State Relationships, Int. J. of Spray and Combustion Dynamics, 1: 339-364, 2009.
- [20] K.T. Kim, a, J.G. Lee, B.D. Quaya and D.A. Santavicca, *Response of partially premixed flames to* acoustic velocity and equivalence ratio perturbations, Combustion and Flame, 157: 1731-44, 2010.
- [21] John P. Longwell, Edward E. Frost, Malcolm A. Weiss Flame *Stability in Bluff Body Recirculation*

Zones Industrial & Engineering Chemistry 1953 45 (8), 1629-1633

- [22] A. Kempf, R.P. Lindstedt, J. Janicka, Large-eddy simulation of a bluff-body stabilized nonpremixed flame, Combustion and Flame, Volume 144, Issues 1-2, January 2006, Pages 170-189
- [23] Stephen, R., Turns, An *Inroduction to Combustion*, Mc Graw Hill, Inc, 1996
- [24] T, Poinsot, and D. Veynante, *Theoretical and Nuimerical Combustion*, R.T. Edwards, Inc.; 2 edition (January 31, 2005)
- [25] ANSYS CFX Mannual and Theory Guide, ©Ansys Inc.
- [26] Culick, F., Lin, W., Jahnke, C., and Sterling, J., Modeling for active control of combustion and thermally driven oscillations, Proceedings of the American Control Conference, 1991
- [27] Kabiraj, L., Suarabh, A., Wahi, P., and Sujith, R., Experimental studies of thermoacoustic instability in ducted premixed flames: Periodic, quasi-periodic and chaotic oscillations, Intl' Summer School and Workshop on Non-Normal and Nonlinear Effects in Aero and Thermocacoustics, Technische Universitat Munchen, 2010
- [28] Kok, J.B.W., and Bram de Jager., *The acoustic wave propagation equation in turbulent combusting flow*, ICSV 14, Carins Australia 2007.